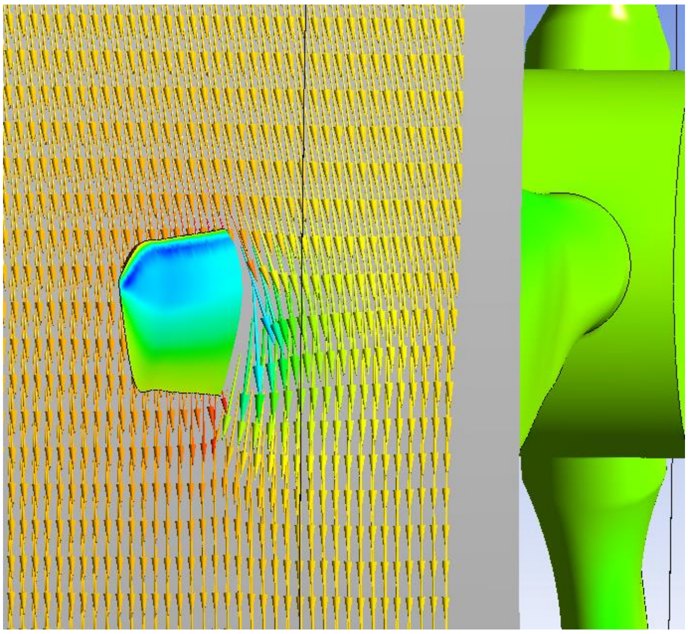
TU Clausthal

Institut für Technische Mechanik

Masterthesis

## Optimization of a wind turbine rotor using CFD



The background of the present project is to study the performance of a wind turbine using Computational Fluid Dynamics (CFD). The work is part of the lecture “Numerical Simulation in Engineering” and shall provide a reference solution for future student assignements.

A prerequisite for the calculation of flows using CFD is the definition of the geometry of a technical object. This is to be done in parameterized form in the present work, so that the geometry can be modified by means of a few control variables and thus the effect of geometry changes on the performance of the system may be investigated. The geometry definition, the simulation setup and post-processing is embedded In the ANSYS workbench. The geometry is defined using SpaceClaim and python.

The result of the work shall be a program that allows the creation of a parametrized 3D representation of the surface of a typical wind turbine profile. Subsequently, the workflow of a CFD simulation is demonstrated, starting with the mesh generation, setting up and running a stationary CFD simulation in ANSYS CFX as well as evaluating the results. The latter shall include the calculation of the power coefficient of the windturbine.

The work on this topic requires a thorough familiarization with ANSYS CFX, in particular the use of scripts in ANSYS SpaceClaim. This requires profound knowledge of programming with python. A sound background in fluid mechanics in general and of computational fluid dynamics is required.

| | |
|---|---|
| Supervision: | Prof. Dr.-Ing Gunter Brenner |
| Place of Work: | ITM/TU Clausthal |
| Commencement date: | as soon as possible |
| Contact: | 05323 72 2515 or gunther.brenner@tu-clausthal.de |